



Propulsion System Dynamic Modeling of the NASA Supersonic Concept Vehicle for AeroPropulsoServoElasticity

NASA Fundamental Aeronautics – High Speed Project - AeroServoElasticity



**George Kopasakis
Joseph Connolly & Jonathan Seidel**

NASA Glenn Research Center
Cleveland Ohio

**50th AIAA Joint Propulsion Conference
July 28 – 30, 2014
Cleveland, OH**

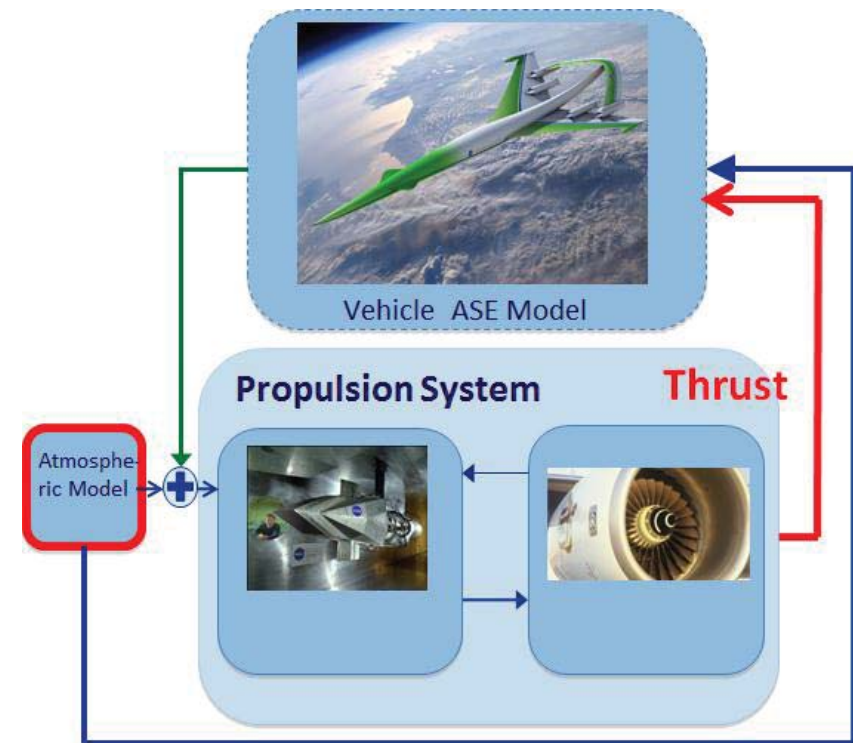


Outline

- **Overview**
- **Propulsion system component modeling**
- **APSE Integration**
- **Conclusions**

AeroPropulsoServoElasticity (APSE) Goals

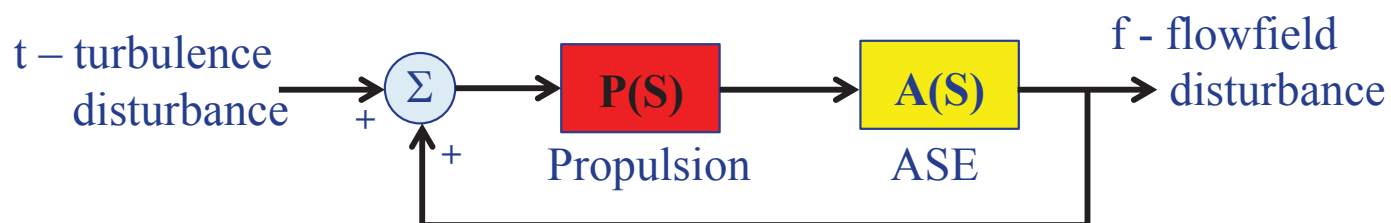
- Develop dynamic propulsion system and aero-servo-elastic/aerodynamic models, and integrate them together with atmospheric turbulence to study the dynamic performance of supersonic vehicles for ride quality, vehicle stability, and aerodynamic efficiency.
- Supersonic vehicles are slender body with more pronounced AeroServoElastic (ASE) modes, which can potentially couple with propulsion system dynamics to present performance challenges.
- Approach for Propulsion System:
 - Develop 1-Dimensional (1D) component models and 2D models where appropriate to be comparable in frequency range to ASE models.
 - Integrate Propulsion with ASE to form a closed-loop dynamic system model to study performance.



Integrated APSE Model

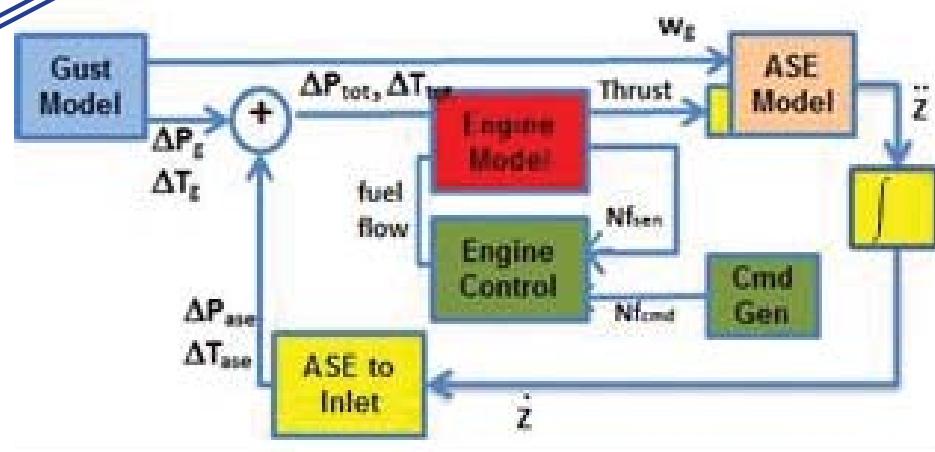
APSE Dynamics - Frequency Range

- ASE modes extend to about 60 HZ when about half the modes are included in the model.
- For that reason propulsion system dynamics need to extend up to approximately 600 Hz in order to also take into account the phase contribution of the propulsion dynamics for the closed-loop system.



$$\frac{F(s)}{T(s)} = \frac{P(s)A(s)}{1 - P(s)A(s)}$$

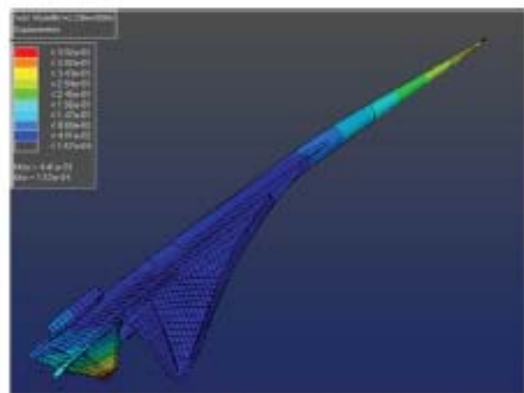
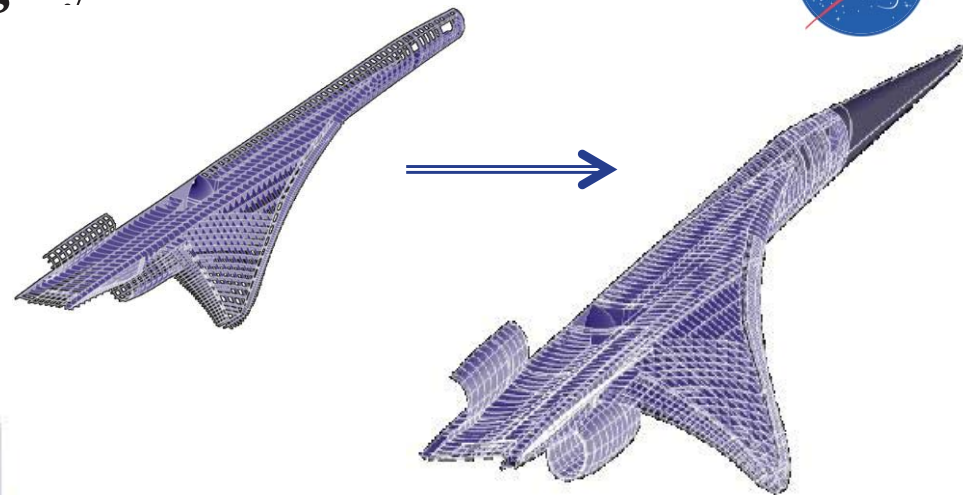
Simplified Closed-loop representation of APSE dynamic coupling



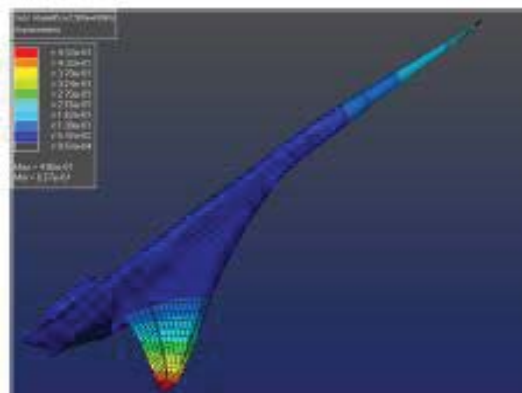
Closed-loop diagram of APSE simulation

ASE Dynamic Model – Langley Research Center

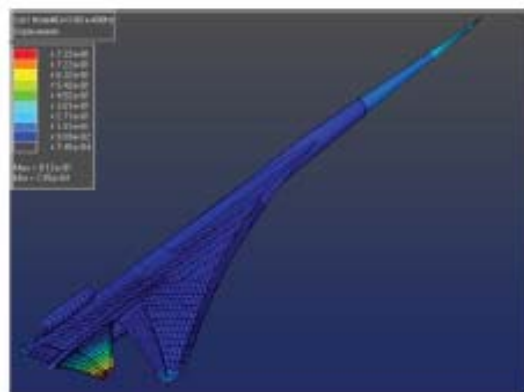
- Constructing 3D models to analyze ASE modes and assess flutter conditions.
- Utilizing 3D model to develop state space models to integrate with propulsion system.



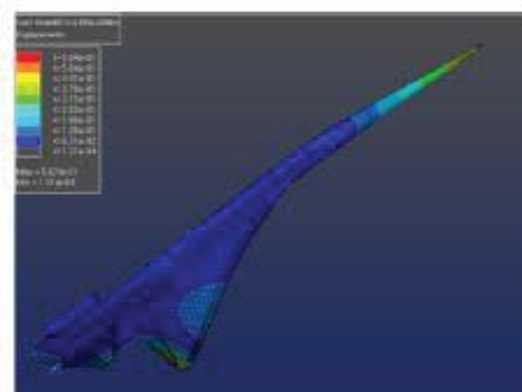
(a) Mode 4, Fuselage Bending, 2.24 Hz



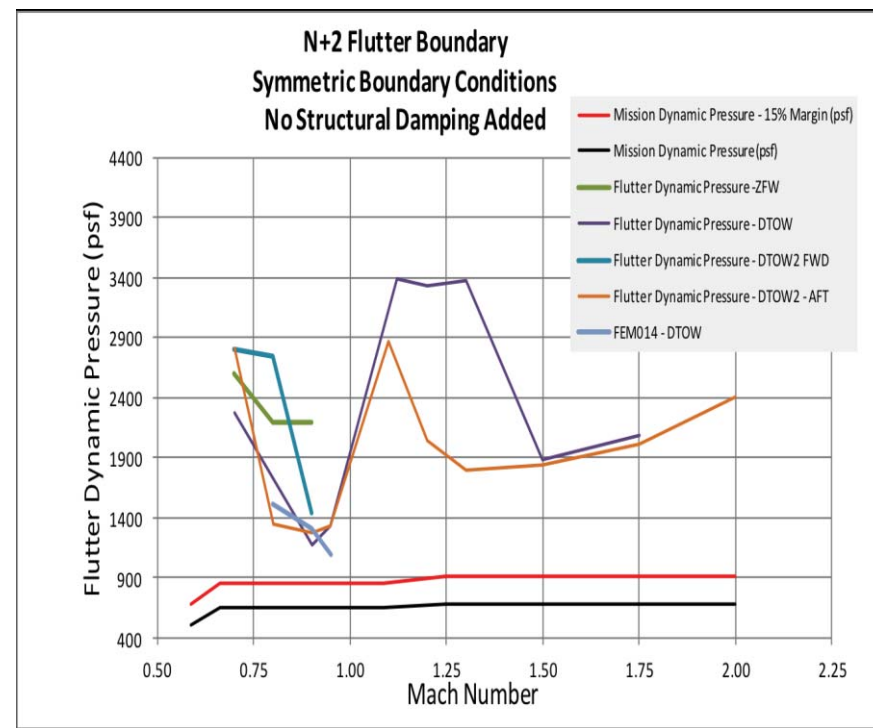
(b) Mode 5, Wing Bending, 2.50 Hz



(c) Mode 6, Tail Bending, 3.06 Hz

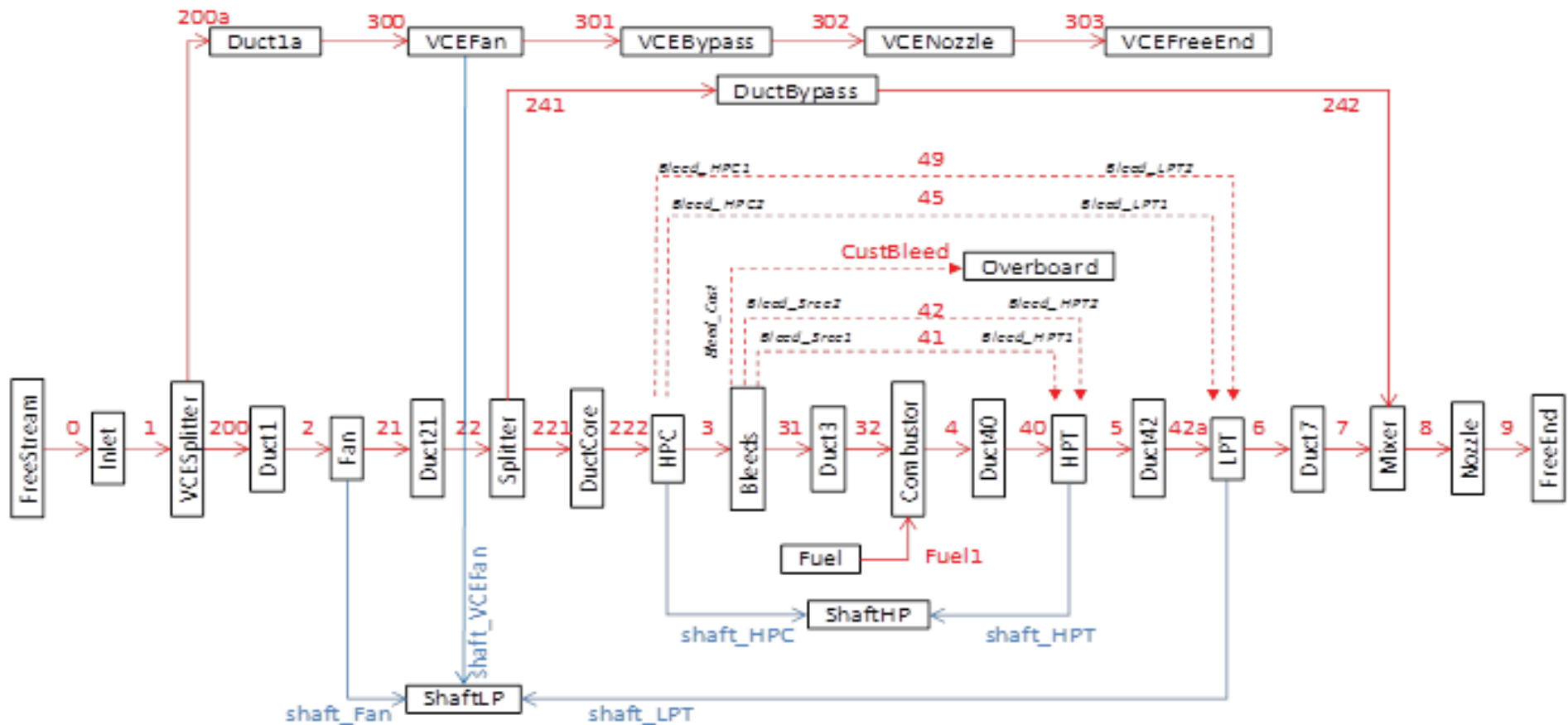


(d) Mode 7, Wing Tip Bending-Torsion-Fuselage 2nd Bending, 4.46 Hz



Propulsion System Dynamic Modeling

- Modeling all the components with lump volume dynamics and performance characteristics and adding combustor and shaft dynamics and variable geometry (Inlet Guide Vanes).
- Model variations include stage-by-stage for compressors and turbines, quasi-1D CFD for the inlet and nozzles, and parallel flow path modeling.



Variable Cycle Engine (VCE) Model

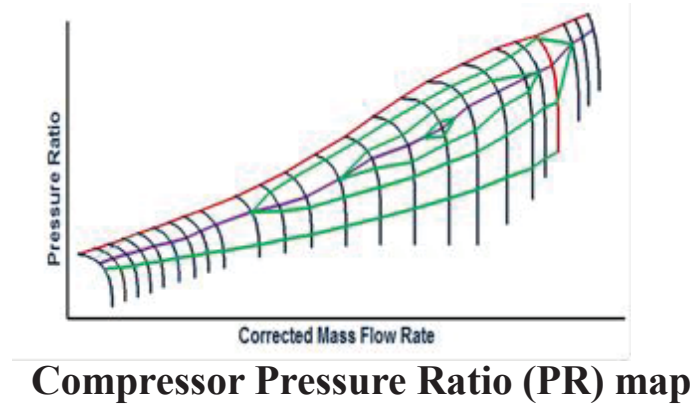
Lump Volume and Stage-by-Stage Component modeling

- Lump volume modeling treats the component as a single volume for conservation dynamics and adds performance characteristics where appropriate

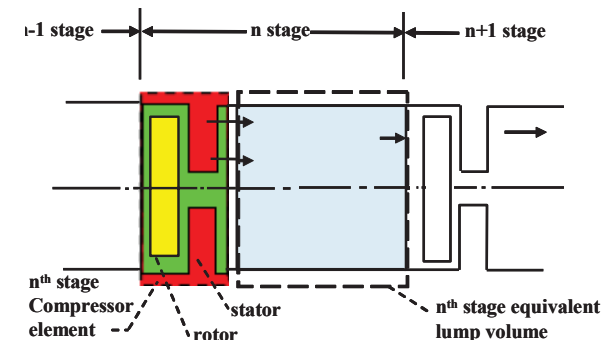
$$\frac{d}{dt}(\rho_{sv}) = -\frac{1}{V} \Delta \dot{W}$$

$$\frac{d}{dt}(\dot{W}) = -\frac{A}{l} \Delta P_{iv}$$

$$\frac{d}{dt}(\rho_{sv} T_{iv}) = -\frac{\gamma}{V} \Delta(T_{iv} \dot{W})$$

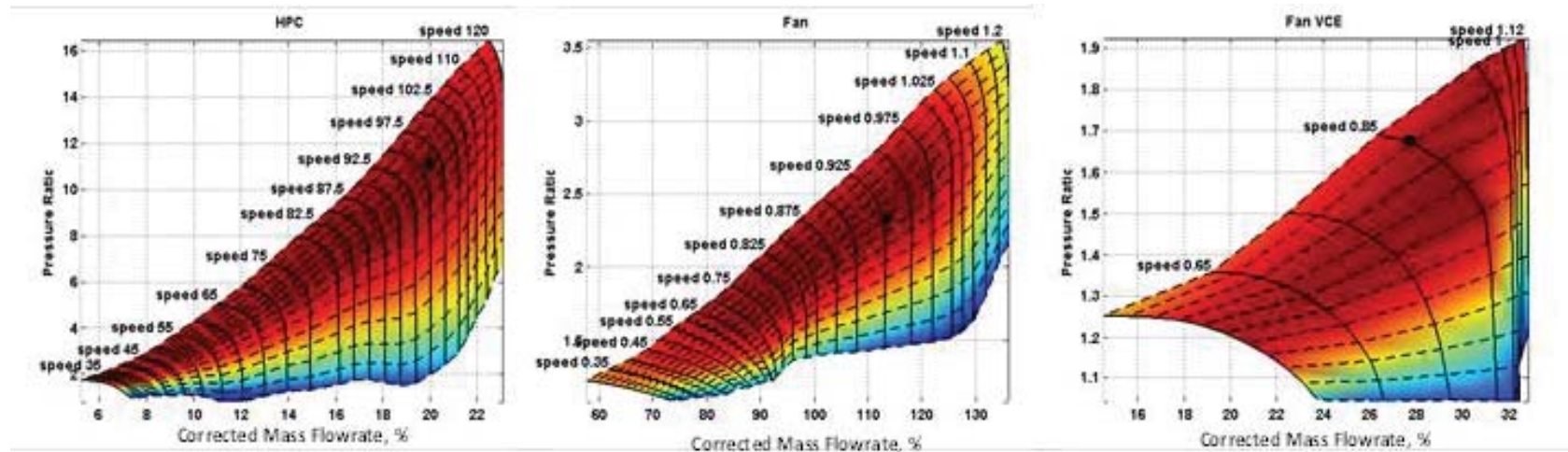


- Stage-by-stage (sbs) modeling treats each stator-rotor pair as a separate dynamic volume with its own performance characteristics
 - The reason this for SBS modeling is higher fidelity, but also for dynamic accuracy (comparing the wavelength of highest frequency accuracy desired to the component acoustic wavelength).
 - Procedure involves scaling performances and successively reducing the component size until the model for the final stage size is developed.
 - Stages are stacked one at a time by successively calculating and adjusting stage areas to match stage delta Mach numbers.



Inlet Guide Vane (IGV) Modeling

- Models compressor variable geometry by incorporating compressor performance characteristics (maps) for different Guide Vane angles



High Pressure Compressor (HPC) map, Fan Map, and VCE Fan map at 0° IGV angles

- IGV dynamics are modeled by calculating the effective compressor area due to IGV, which effects conservation dynamics

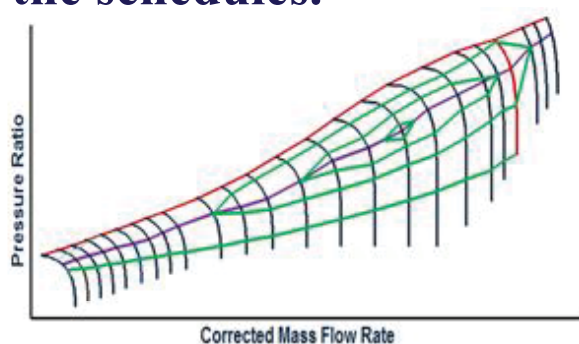
$$A = \frac{\dot{W} \sqrt{T_t}}{MFP * P_t \cos(\alpha)}$$

$$MFP = M \sqrt{\frac{\gamma g}{R}} \left(1 + \frac{\gamma-1}{2} M^2 \right)^{\frac{\gamma+1}{2(1-\gamma)}}$$

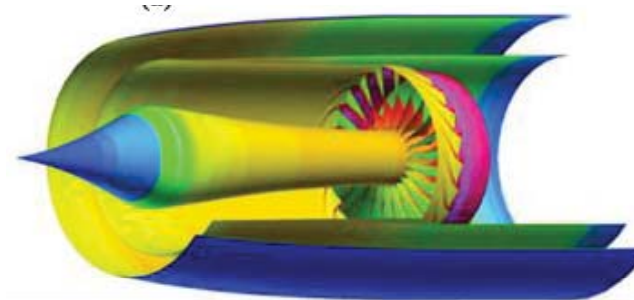
Engine Control Schedules



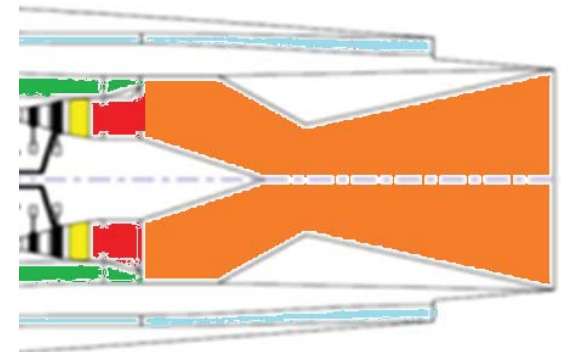
- Process has been developed to derive engine control Schedules utilizing the dynamic engine model.
- For the compressor the engine is run at various operating speeds and then
 - a) a plug nozzle utilizing the compressible choked flow equation is developed to provide the same mass flowrates for the compressor.
 - b) The IGV angle is changed to device a schedule to operate the compressor at the desired operating line on the PR vs. CFMR map.
- For the Inlet the centerbody is translating and/or collapsed to position the normal shock by the cowl-lip to minimize mass flow spill and associated drag.
- Given an A8 area for desired engine flow demand and thrust, the nozzle area is trimmed to device a schedule to fully expand the flow.
- The derivation of these schedules is an involved and laborious process. So lately the Numerical Propulsion System Simulation (NPSS) has been utilized instead to derive the schedules.



Compressor Pressure Ratio (PR) map



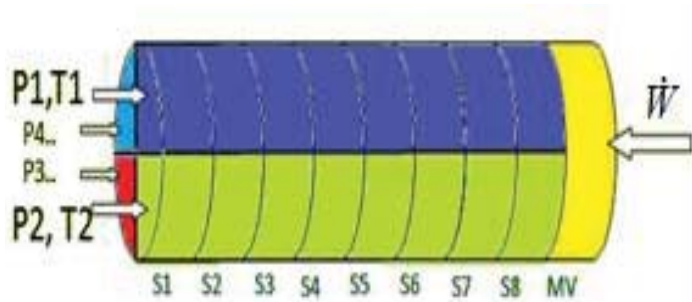
Inlet



Nozzle

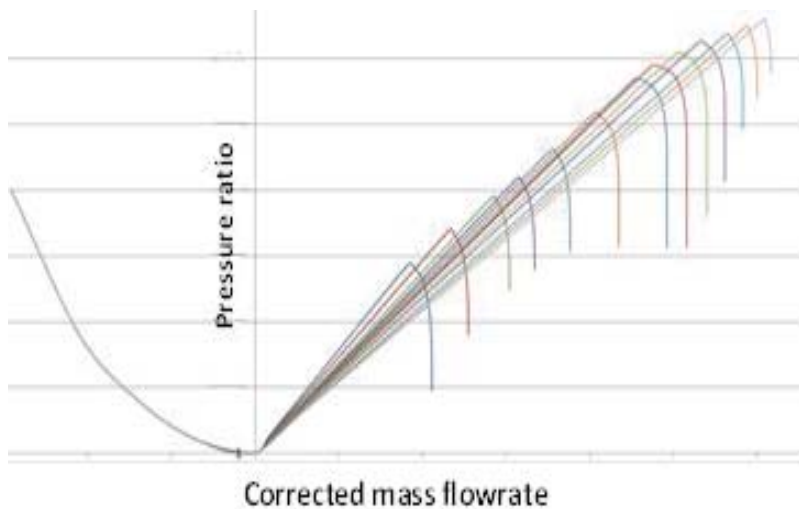
Compressor Parallel Flow Path modeling

- **Objective:** To capture dynamics of flow distortion and dynamics of surge-stall
- **Approach:** Simulate both axial and rotational flow, since stall event starts with a stall cell at some compressor stage. A procedure has been developed.

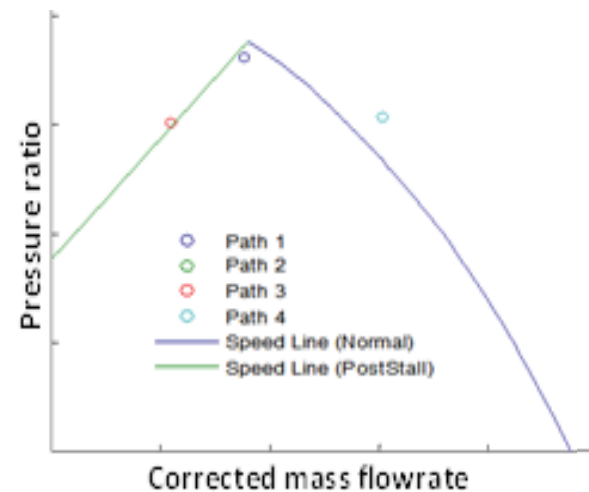


$$\frac{\partial}{\partial t}(W_j) = -a_{xj} \frac{\partial}{\partial x}(F_{xj}) - a_{\phi j} \frac{\partial}{\partial \phi}(F_{\phi j}) + S_j$$

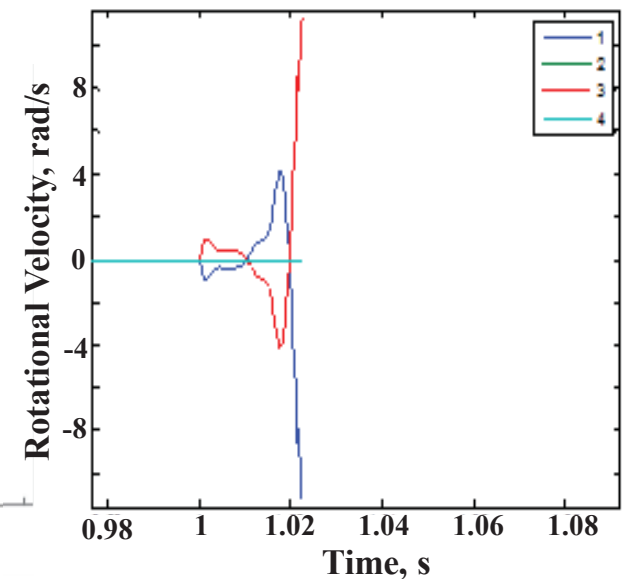
W_1 - mass
 W_2 - axial momentum
 W_3 - rotational momentum
 W_4 - energy



Typical PR map for a certain stage path



Shows parts of stage 7 in the stall region (snapshot)



Rotational velocities of stage 7 during stall

- Similarly develop parallel flow path models for rest of components upstream of combustor



Combustor, Turbine, Rotors, and Duct modeling

- The combustor as reported before is modeled by the combustion dynamics of fuel transport-mixing, flame, and acoustics and respective volume dynamics with heat addition, combustion efficiency, and pressure drop.

ex. fuel, flame & acoustic dynamics:

$$\frac{\dot{W}_f''}{\dot{W}_f} = \frac{K e^{-\tau_t s}}{(\tau_f s + 1)(\tau_a s + 1)}$$

- Turbine also modeled as reported before with volume dynamics and performance characteristics, but with 2 spools and engine specific cooling flows.
- A stage-by-stage turbine modeling process has been developed, which is similar to the compressor stage-by-stage modeling.
- Rotor model same as before except in this case there are 2 rotors.

$$\frac{d}{dt}(N) = \frac{K_N}{N} [\Delta E_{turb} - \Delta E_{comp}]$$

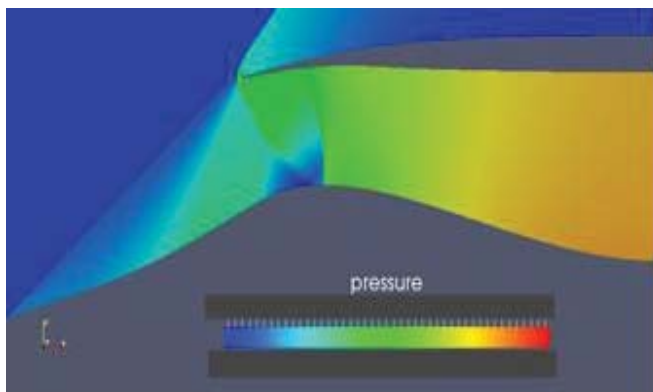
- Ducts are modeled simply by utilizing volume dynamics.

Kopasakis, G.; Connolly, J. W.; Paxson D. E.; Ma P.; "Volume Dynamics Propulsion System Modeling for Supersonic Vehicle Research," GT2008-50524, ASME TurboExpo 2008, Berlin Germany; NASA/TM 215172; Journal of Turbomachinery, Vol. 132, October 2010.

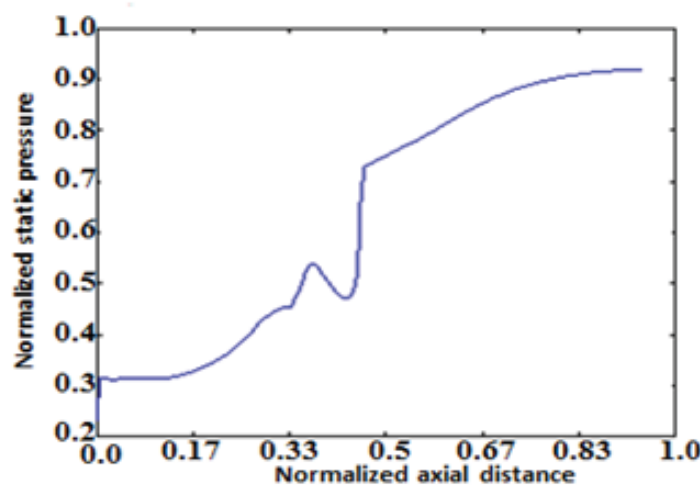
Connolly, J.; Kopasakis, G.; Paxson, D.; Stuber, E.; Woolwine, K.; "Nonlinear Dynamic Modeling and Controls Development for Supersonic Propulsion System Research," 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, AIAA 2011-5635, August, 2011.

Inlet Dynamic Modeling

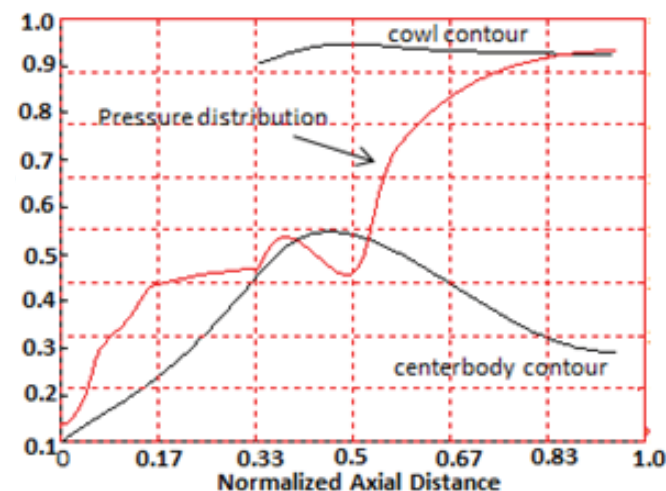
- Initially, a quasi 1D dynamic inlet model called NOIMA (Nonlinear Inlet Modeling Assembly) was developed for mixed compression type inlets.
- This model with the help of test data was extended to model the test inlet and subsequently the concept N+2 external compression inlet utilizing quasi 1D CFD with leakage fluxes and moving computational domains.
- Separately in a graduate grant (PhD.) with the University of Colorado 3D & 2D CFD is utilized to model this inlet and derive dynamic factors to help develop a quasi 1D dynamic model.
- Characterizing inlet dynamic accuracy with frequency domain responses is in process.



2D CFD pressure flow field



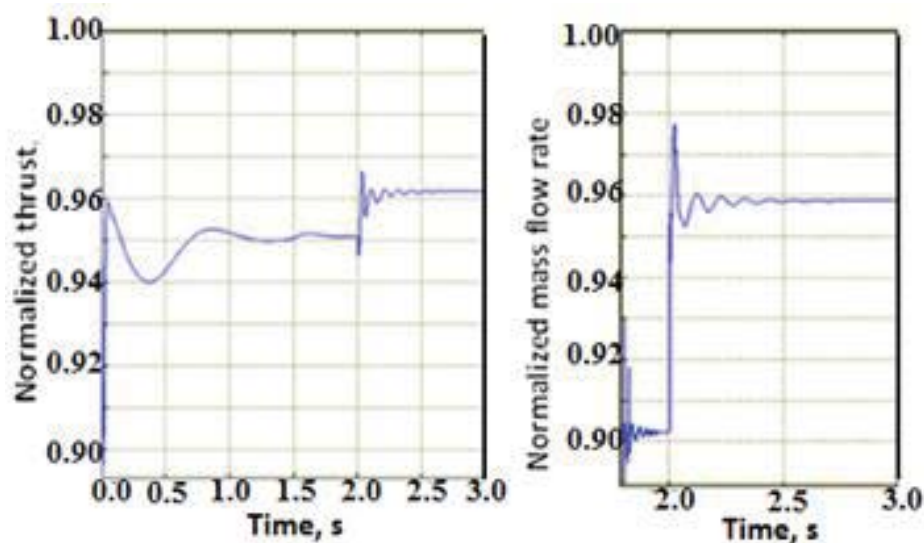
Pressure axial distribution from 2D



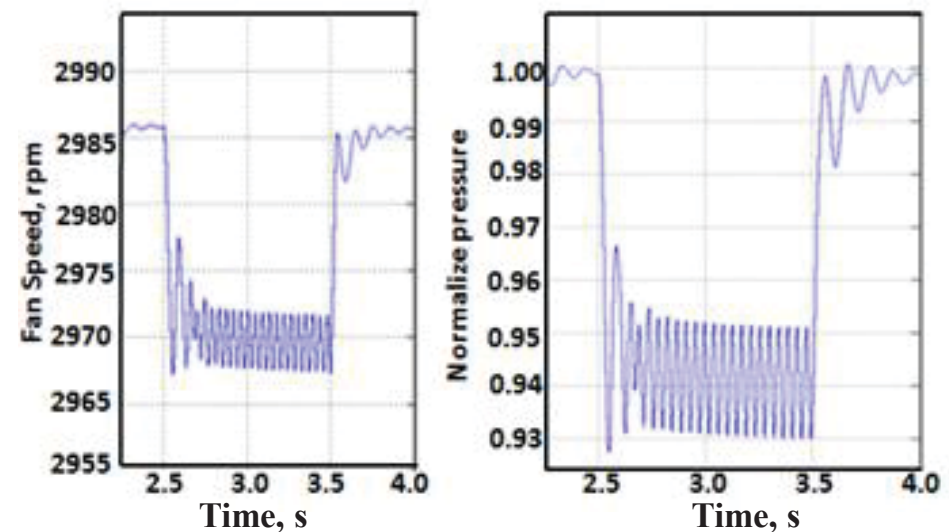
Pressure axial distribution from quasi 1D CFD

Inlet-Engine Integration

- The N+2 inlet engine face boundary conditions modified to match those of the VCE and the two were integrated.
- Performance of the integrated simulation were about as expected, albeit the simulation presently runs much-much slower in SIMULINK due to the fast inlet dynamics compared to the slower engine dynamics that takes longer time to reach steady state.



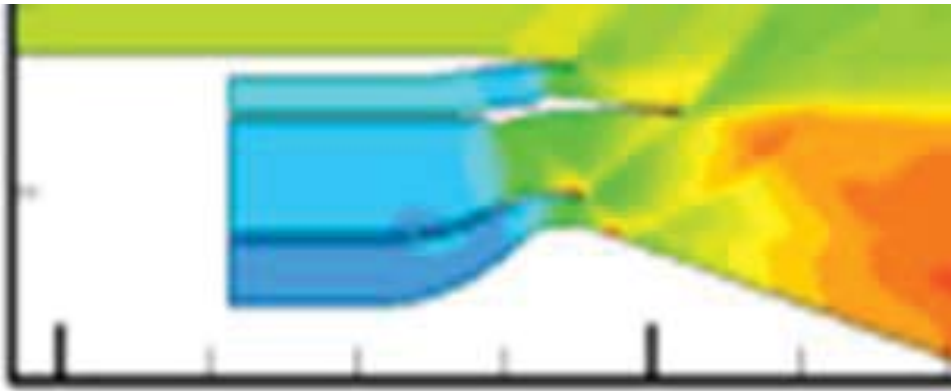
Engine thrust and mass flowrate (engine start at 0 sec, inlet starts at 1.8 sec, the two are switched to run together at 2 sec)



Engine speed and engine face pressure due to steps in fan speed command

Nozzle Dynamic Model

- Initially the compressible flow – choked flow relation was utilized to model the nozzle as described in references.
- Later the McCormack method was used to develop a quasi 1D CFD model of the internal nozzle passages and compared to CFD simulations utilizing the CESE method.
- Objective is to find minimum grid resolution required to accurately capture the nozzle dynamics, which so far is found to be orders of magnitude less than the grids typically required for steady state accuracy



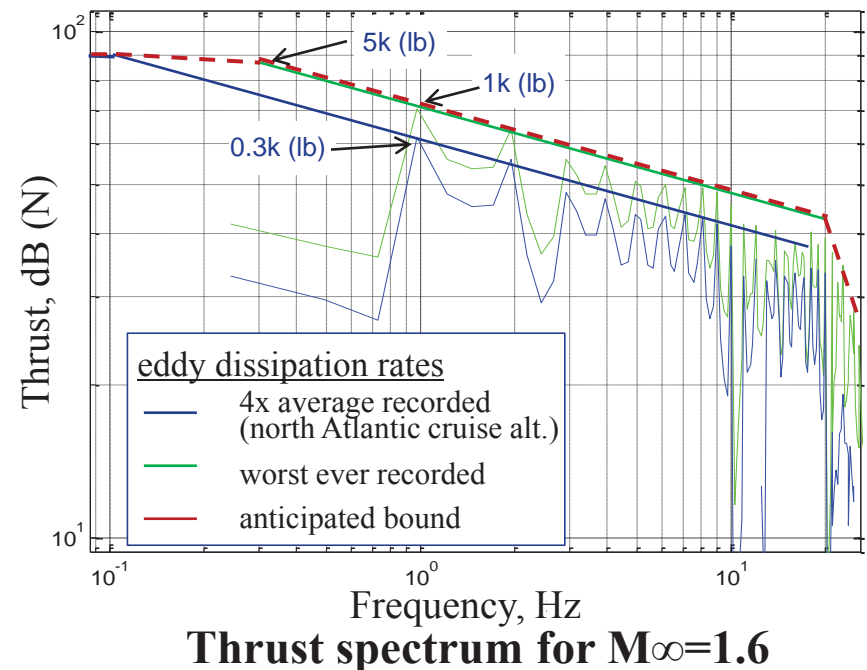
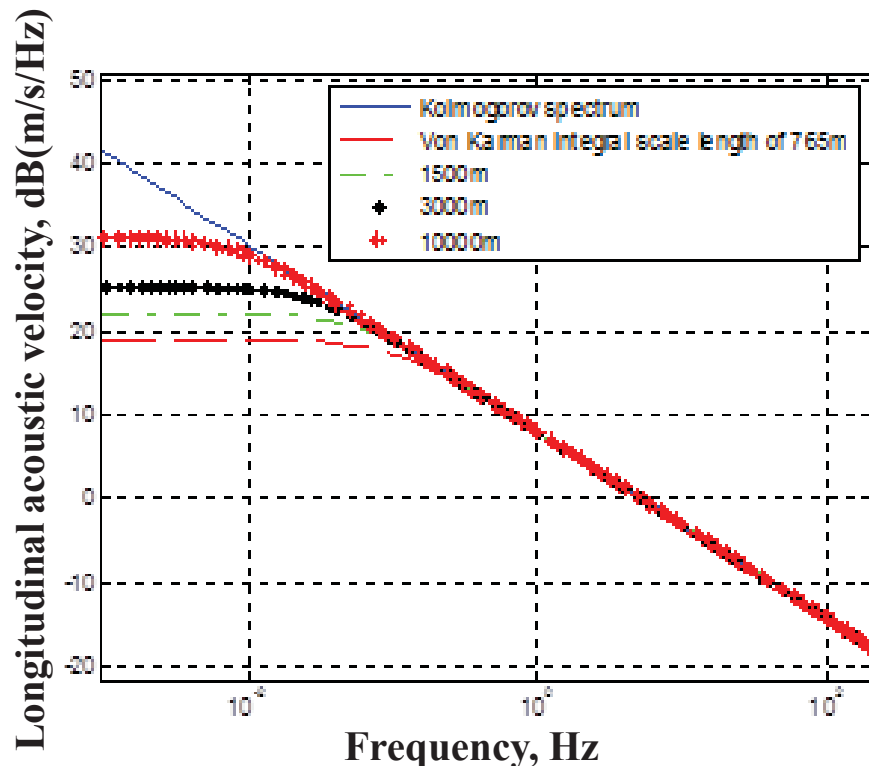
Nozzle CFD simulation



Atmospheric Turbulence Model

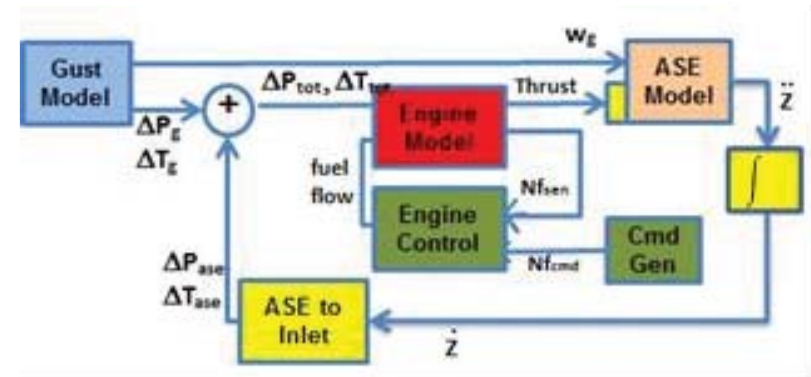
- Atmospheric turbulence model developed based on fractional order fits to provide more accuracy than the traditional Dryden model, especially for high speed vehicles.
- Model provides freestream flow field disturbance to propulsion system and gust loads to vehicle structure.
- Approach develops frequency domain filters based on atmospheric turbulence parameters to filter unit amplitude spectral sinusoids that provide the appropriate disturbance excitations of acoustic gusts, temperature, and pressure.

ex. Filter: $G_{LA}(s) = 70e^{2/9} \frac{(s/9.2+1)(s/55.0+1)(s/335.5+1)}{(s/1.46+1)(s/30.1+1)(s/85.7+1)(s/1593.1+1)}$

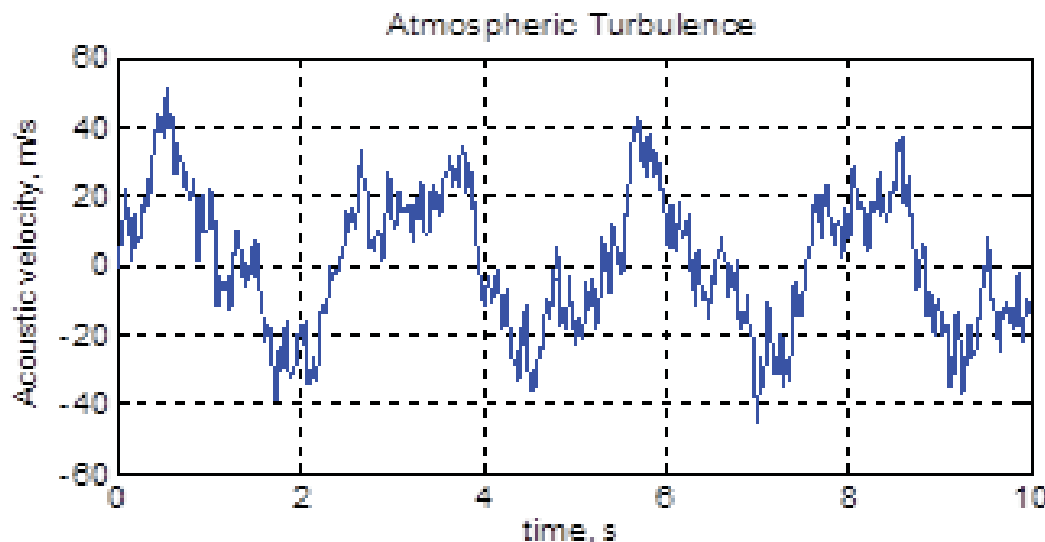


APSE Integration

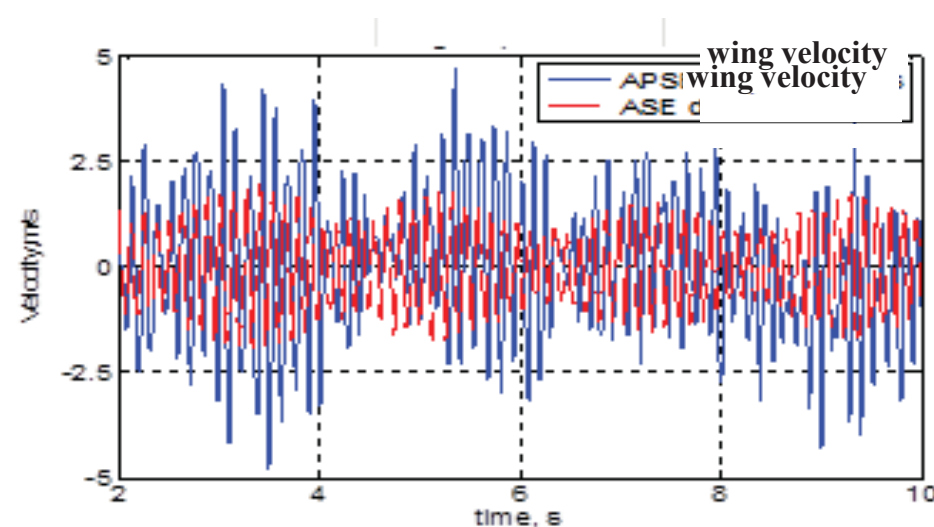
- First closed-loop APSE simulation developed consisting of a single engine and a semi-span wing (LaRC & GRC). Future plans to simulate whole vehicle.
- Interfaces from turbulence to propulsion and ASE to propulsion recently updated and covered in the paper
- Thrust to ASE interface, new to ASE models is in the process of been updated utilizing recent testing of applying forces to the engine pylons and measuring acceleration along the wing



Closed-loop diagram of APSE simulation



Longitudinal wind velocity due to pure wind gusts and temperature generated wind gusts



Wing velocity at vicinity of engine inlet for APSE system and for ASE alone

Conclusions



- **APSE forms a closed-loop dynamic system and thus the modeling of this system needs to take into consideration the closed-loop feedback system dynamics.**
- **Propulsion system component models have been developed based on these guidelines and additional type models are being developed to address the effect of flow distortion.**
- **First integrated closed-loop APSE model developed with the future goal of developing a complete dynamic vehicle model to address performance such as vehicle stability, ride quality, and aerodynamic efficiency.**
- **Additionally, this model provides the capability to understand the controls improvements required to suppress potential APSE dynamic couplings like flutter, to design integrated flight controls, and the atmospheric conditions for safe flight.**